Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 82 (2015) 258 – 264
Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

# Numerical Study of Compressor Fouling Mechanism based on Eulerian–Eulerian Approach

Keyvan Shaabani Lakeh[a]*, Alberto Martinelli[a], Augusto De La Torre[a], Gianluca Montenegro[a], Angelo Onorati[a]

[a]*Internal Combustion Engine Group, Department of Energy, Politecnico di Milano, Via Lambruschini 4a, 20156,Milan, Italy*

## Abstract

The paper describes a fluid dynamic numerical model to study the fouling mechanism for turbomachines (such as large gas turbines and small turbochargers). For this purpose, the gas-particle behaviour over a compressor blade is modelled using an open source CFD tool, OpenFOAM®. Through this model, two-fluid implicit equations system, with Eulerian-Eulerian approach is considered. This approach uses RANS turbulent models and also takes into account the particle fluid interactions. Applying Johnson-Kendal-Robert (JKR) theory, the particle deposition over the blade surface is predicted. According to the simulation results, there are certain regions which are facing the huge amount of deposition. The surfaces of these regions are prone to form an almost thick layer of sticked particles and affect the boundary layers and blade aerodynamics significantly. In order to capture this effect, a model to predict the deformation of the boundary with respect to deposition rate is also introduced.


Peer-review under responsibility of the Scientific Committee of ATI 2015

*Keywords:* Compressor blade fouling; Particle deposition; OpenFOAM; Eulerian-Eulerian two phase model; Mesh motion

## 1. Introduction

There are various mechanisms for deterioration in gas turbines. Among them, the four main mechanical degradation mechanisms are: fouling, erosion, damage, abrasion [1]. Fouling phenomena is a major cause of performance deterioration in turbomachines. Mostly, it results in a significant reduction in efficiency of compressors. Fouling is the process of contamination of the surfaces with the particles transported by the working fluid. They form a layer over the surface, which will increase the surface roughness and change the aerodynamics of the blades, resulting in a significant decrease in performance. The deposition rate can be affected by size, amount, and chemical nature of the aerosols [2].

* Corresponding author. Tel.: +39-380-7798034.
*E-mail address:* keyvan.shaabani@polimi.it


Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.12.031

The development of fouling models in turbomachinery has been a concern for several decades. Based on particle entrainment mechanism developed Fuks [3]. Tarabrin [4] proposed an analytical integral model using geometrical entrainment. They calculated deposition rate as a function of bulk stokes number and geometry for the axial compressors. According to widespread development of numerical calculation methods in fluid dynamics, various attempts have been performed to numerically model the deposition process in particle-entrained flow problems.. The key point in the models is to capture the phase interference in momentum equation. The first models considered the drag force as the main phase intervention term in momentum equation, and solve the governing equations defining volume fraction. More sophisticated models have been developed through the time both in Lagrangian [5-7] and Eulerian [8-12] approaches, encountering all the possible interferences to be applied to momentum equation in the terms of virtual mass force, drag force, lift force, Basset force, etc. Applying these terms make us to solve the momentum equations implicitly that will significantly increase the complexity of the problem. However, a more simplified approach assumes the dilute particle-laden flow, which ignores the effect of particle portion on the main fluid dynamics [13, 14]. Through this paper a two-fluid model is used with Eulerian-Eulerian approach. The solver is already implemented in the open-source software, OpenFOAM®. In this work, the solver is modified in a way to be able to predict the rate of deposition on the surface as well as the resulted surface deformation through the time.

**Nomenclature**

| | |
|---|---|
| $\alpha$ | Phase volumetric fraction |
| U | Velocity (m/sec) |
| $\rho$ | Density (kg/m$^3$) |
| g | Gravity acceleration (m/sec$^2$) |
| p | Pressure (Pa) |
| $\theta$ | Granular Temperature (m$^2$/sec$^2$) |
| $\gamma$ | Specific surface energy (kg/sec$^2$) |
| R | Average radius of the particles (m) |
| K | Spring factor of the elastic body |

## 2. Numerical Method

Through this research a particle deposition model is added to the solver to calculate the volumetric rate of deposition at boundary cell faces. According to the deposition amount, the surface will be displaced. Therefore, a solver is developed to calculate and apply the change in geometry to the solution process.

### *2.1. Governing Equations*

To capture the fouling effect a two-fluid model is used. Distributed particles are considered as a secondary continuum phase [15], while the model tries to captures the phase interference and related changes in governing equations. For such a fluid system a volume fraction can be defined for each phase. Therefore, continuity should be conserved for each phase separately. In contrary, the momentum equation is changed according to the interaction between two phases. This interphase momentum transfer term appears as a source term for each phase (see equation 1).

$$\frac{\partial \alpha_\varphi \bar{U}_\varphi}{\partial t} + \nabla.\left(\alpha_\varphi \phi_\varphi U_\varphi\right) + \nabla.\left(\alpha_\varphi \bar{R}_\varphi\right) = -\frac{\alpha_\varphi}{\rho_\varphi}\nabla \bar{p} + \alpha_\varphi g + \frac{\bar{M}_\varphi}{\rho_\varphi} \quad (1)$$

The momentum transfer includes various interaction phenomena in terms of force [16]. Drag and lift force, virtual mass momentum, and Basset force are the major terms [17]. Gosman [18] claimed that only drag force has significant effect. However, in current research the other sources are taken to account.

The turbulence model is also based on the two-fluid assumption. There are main reasons why the model deviates from the standard one fluid model, such as random stirring in the flow made by particles, the wake produced behind particles, or particle intervention in turbulent eddies. Gosman [18] proposed a two-phase turbulence model consisting of the standard $k-\varepsilon$ model by solving the phase-combined transport equations for k and ε. In another approach Van Wachem [19] used gas kinetic theory to describe the phase stress produced by kinetic and collisional contributions separately for particle phase in solid-gas flows. The original work is produced by Lun [20]. In this method the Granular temperature Θ is defined to express particle fluctuations [19].

$$\Theta = \frac{1}{3}|{U'}_p^2| \tag{2}$$

According to this definition, the granular energy and the dissipation rate transport equation can be also derived as a function of Θ [19]. To extract the pressure flow properties such as solid pressure and viscosity as a function of Θ several models have been developed [20, 21]. In this research, the multiphase turbulence model is selected based on the kinetic theory approach [20].

### *2.2. Deposition Model*

The deposition is a stable state of sticking particles in contact with the surface. The margins of situation, in which the particles do not leave the surface after impact, is revealed by the contact mechanics theory. In this study the Johnson-Roberts-Kendall model is used to predict the behavior of the particles after meeting the turbomachine surface [22]. Thornton and Ning [23] used JKR model to calculate the critical impact velocity (see equation 3). The particles that hit the surface with higher velocities will be bounced back, while lower velocity guaranties the sticking to happen.

$$U_s = 1.67\left(\frac{K^2\gamma^5}{\rho_p^3 R^5}\right)^{1/6} \tag{3}$$

This deposition model is also used in similar studies [24, 25] but with Lagrangian approach.

### *2.3. Mesh motion model*

The mesh moving model consists of a standard transport equation written on relative velocity of the flux respect to boundary patches. The mesh moving must satisfy the space conservation law and preserve the spatial consistency. However, the proper equation (Piola-Kirchhoff) is nonlinear and too heavy to solve. To increase the velocity of the solver we can introduce two simplified equations. The first equation is Laplacian scheme. The other equation is Linear Pseudo-Solid (LPS) scheme. Both methods introduce some errors, because the conservation law in these cases isn't exactly satisfied. The Laplacian and the LPS methods are suggested for irregular deformation of the boundaries, such as fouling problems [26]. The calculation process starts from boundary's point displacement, which must be provided. Then, the solver will make the mesh smooth using one of the two equations. In OpenFOAM, the Laplacian scheme for motion solver has been implemented to diffuse the boundary motion to the internal mesh points. The boundary points are displaced in order to reproduce the growth of the deposited layer. This boundary motion is then diffused into the internal points with a diffusivity coefficient which is inversely proportional to the point distance from the moving boundary. To efficiently exploit the numerical framework of the code, cell based displacement is solved and then the solution is interpolated on the points. This usually leads to efficient and fast solution of the point motion equation, but implies difficulties in controlling the mesh quality when large deformations occur.

## 3. Case study

The case study selected as a two dimensional flow over a blade. Borello, Rispoli and Venturi [24] modeled the same case in 3D with Eulerian-Lagrangian approach. The original Geometry related to a study on a cascade containing blades of GE rotor B experimentally [27].

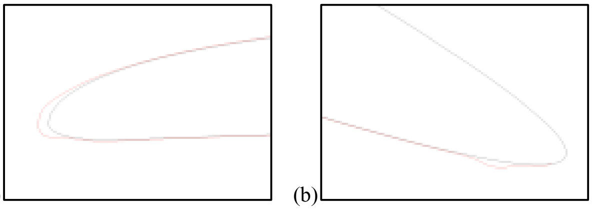
Solution in two dimensions allows us to avoid mesh related complications. The mesh contains about 20k structured cells (figure 2). The refinement process in this case is restricted by thickness of deposition which will increase the possibility of divergence in mesh motion solver. Therefore, the cells at the boundary were designed in a way that they would be far bigger than the amount of deposition thickness, and at the same time they had to be fine enough to capture fields inside the boundary layer. The mesh was created by automatic commercial software and is structured rectangular. The maximum non-orthogonality is kept under 30 degrees. To form a boundary layer around the blade a simple grading with the ration of 1.1 was used which leads to maximum aspect ratio of 6.8.

Fig. 1. Generated 2D mesh for GE rotor B single blade

A PISO algorithm with adequate non-orthogonal correctors was selected to obtain the acceptable solution. The solver uses PISO algorithm for each phase and solve the system of equations implicitly. To predict the turbulence term, the Kinetic Theory model developed by Lun [20] is used. A particle-laden air flow with constant subsonic speed (U=25 m/sec) passes through the blades horizontally (similar to original experimental study). The effect of different dust concentration and change in fluid velocity are investigated. To model the contact mechanics the density of the particles are considered $1200 kg/m^3$, the adhesion energy to $0.12 kg/m^2$ [24], the overall spring factor $9 \times 10^{-11}$, and the particle average diameter to 1 micron.

## 4. Results and Discussion

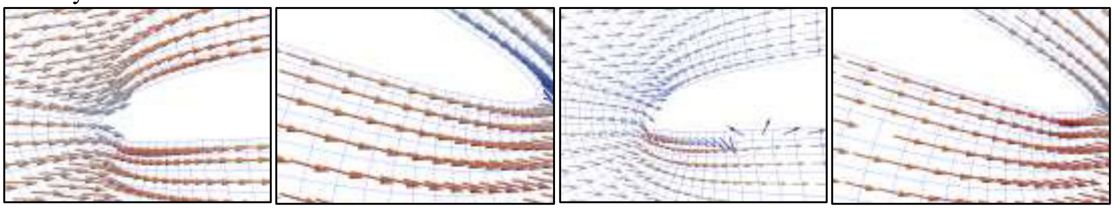
It is expected to observe the deposition film on the surface of the blade mainly at two spots. At the leading edge when particles directly hit the surface and in the tail of the blade at pressure side when according to the tail wake we will have a huge gradient (deviation) for the main flow (figure 3). Looking at the results reported in [24] at the center of the blade's height (to avoid the secondary vortex flow at hub as well as tip clearance effects) the same result is reported.



(a) (b)

Fig. 2. The deposition at leading edge(a) and tail (b). red lines show the deposition and the black lines show the original surface.

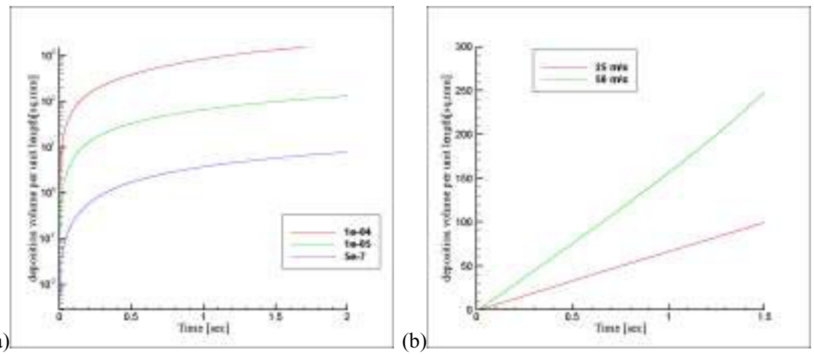
As stated above, this change in profile shape will influence the flow aerodynamics around the blade. Figure 4 reveals the effect of deposition formation on the boundary flow over blade at leading edge and blade tail. The steady state results without fouling are also shown. The comparison between two cases illustrates that not only the resulted surface roughness can change the flow pattern over the surface, but also they can make local instabilities that can influence the flow turbulence behavior.



(a) (b) (c) (d)

Fig. 3. Velocity vectors for steady-state single phase solution at (a) leading edge (b) tail, and the same vectors for the blade under fouling effect after 2 sec (c) leading edge (d) tail.

The effect of the particle concentration is also studied in this research. Figure 5 shows the evolution of fouling layer in terms of volumetric rate around blade profile for different initial particle concentrations. The results show that the dependency of deposition on the concentration is not exactly linear. The nonlinearities might happen according to the change in roughness of the deposited surface as well as alternation in boundary layer flow regime due to phase interaction forces. Also, different velocity studies depicted that increasing in flux will affect the deposition behavior in an almost linear manner. In other word, the effect of the fluid speed on nonlinear terms that affecting deposition is insignificant in comparison with the effect of phase interaction. As the concentration increases the nonlinear terms of phase interaction forces and granular turbulence will also appear more significantly in momentum equations, which lead to nonlinear behavior in deposition process.



(a) (b)

Fig. 4. The volumetric deposition (a) for three different concentrations (b) for two inlet gas velocities

## 5. Conclusion

This study was dealing with implementation of particle deposition model in a two phase Eulerian-Eulerian solver. Also, the effect of the fouling phenomena on the geometry of the blade is modeled by a mesh moving solver. The results show the significant change in flow regime around the blade profile. The fouling films formed at the leading edge and mostly at the pressure side of blade tailing edge, which are also reported by three dimensional models and experimental studies.

## References

[1] Kurz, R. and K. Brun, *Degradation in gas turbine systems.* Journal of Engineering for Gas Turbines and Power, 2001. **123**(1): p. 70-77.

[2] Kurz, R. and K. Brun, *Fouling mechanisms in axial compressors.* Journal of Engineering for Gas Turbines and Power, 2012. **134**(3): p. 032401.

[3] Fuks, N.A., *The mechanics of aerosols*. 1989: Dover Publications, New York, US.

[4] Tarabrin, A., V.A. Schurovsky, A.I. Bodrov, and J.P. Stalder, *An Analysis of Axial Compressors Fouling and a Cleaning Method of Their Blading*. in *ASME 1996 International Gas Turbine and Aeroengine Congress and Exhibition*. 1996. American Society of Mechanical Engineers.

[5] Birello, F., D. Borello, P. Venturini, and F. Rispoli, *Modelling of deposit mechanisms around the stator of a gas turbine*. in *ASME Turbo Expo 2013: Turbine Technical Conference and Exposition*. 2013. American Society of Mechanical Engineers.

[6] Durst, F., D. Miloievic, and B. Schönung, *Eulerian and Lagrangian predictions of particulate two-phase flows: a numerical study.* Applied Mathematical Modelling, 1984. **8**(2): p. 101-115.

[7] Konstandopoulos, A.G., *Deposit growth dynamics: particle sticking and scattering phenomena.* Powder Technology, 2000. **109**(1): p. 262-277.

[8] Young, J. and A. Leeming, *A theory of particle deposition in turbulent pipe flow.* Journal of Fluid Mechanics, 1997. **340**: p. 129-159.

[9] Slater, S., A. Leeming, and J. Young, *Particle deposition from two-dimensional turbulent gas flows.* International Journal of Multiphase Flow, 2003. **29**(5): p. 721-750.

[10] Mohanarangam, K., Z. Tian, and J. Tu, *Numerical simulation of turbulent gas–particle flow in a 90° bend: Eulerian–Eulerian approach.* Computers & Chemical Engineering, 2008. **32**(3): p. 561-571.

[11] Mathiesen, V., T. Solberg, and B.H. Hjertager, *Predictions of gas/particle flow with an Eulerian model including a realistic particle size distribution.* Powder Technology, 2000. **112**(1): p. 34-45.

[12] Enwald, H., E. Peirano, and A.-E. Almstedt, *Eulerian two-phase flow theory applied to fluidization.* International Journal of Multiphase Flow, 1996. **22**: p. 21-66.

[13] Elghobashi, S., *On predicting particle-laden turbulent flows.* Applied Scientific Research, 1994. **52**(4): p. 309-329.

[14] Wang, Q. and K.D. Squires, *Large eddy simulation of particle-laden turbulent channel flow.* Physics of Fluids (1994-present), 1996. **8**(5): p. 1207-1223.

[15] Ishii, M. and T. Hibiki, *Thermo-fluid dynamics of two-phase flow*. 2011: Springer Science & Business Media.

[16] Magnaudet, J. *The forces acting on bubbles and rigid particles*. in *ASME Fluids Engineering Division Summer Meeting, FEDSM*. 1997.

[17] Rusche, H., *Computational fluid dynamics of dispersed two-phase flows at high phase fractions*. 2003, Imperial College London (University of London).

[18] Gosman, A., C. Lekakou, S. Politis, R.I. Issa, and M.K. Looney, *Multidimensional modeling of turbulent two-phase flows in stirred vessels.* AIChE Journal, 1992. **38**(12): p. 1946-1956.

[19] van Wachem, B.G.M., *Derivation, implementation, and validation of computer simulation models for gas-solid fluidized beds*. 2000: TU Delft, Delft University of Technology.

[20] Lun, C., S.B. Savage, D.J. Jeffrey, and N. Chepurniy, *Kinetic theories for granular flow: inelastic particles in Couette flow*

*and slightly inelastic particles in a general flowfield.* Journal of fluid mechanics, 1984. **140**: p. 223-256.

[21] Gidaspow, D., *Multiphase flow and fluidization: continuum and kinetic theory descriptions*. 1994: Academic press.

[22] Johnson, K., K. Kendall, and A. Roberts. *Surface energy and the contact of elastic solids*. in *Proceedings of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*. 1971. The Royal Society.

[23] Thornton, C. and Z. Ning, *A theoretical model for the stick/bounce behaviour of adhesive, elastic-plastic spheres.* Powder technology, 1998. **99**(2): p. 154-162.

[24] Borello, D., F. Rispoli, and P. Venturini, *An integrated particle-tracking impact/adhesion model for the prediction of fouling in a subsonic compressor.* Journal of Engineering for Gas Turbines and Power, 2012. **134**(9): p. 092002.

[25] Bouris, D., R. Kubo, H. Hirata, and Y. Nakata, *Numerical comparative study of compressor rotor and stator blade deposition rates.* Journal of engineering for gas turbines and power, 2002. **124**(3): p. 608-616.

[26] Jasak, H. and H. Rusche. *Dynamic mesh handling in OpenFOAM*. in *Proceeding of the 47th Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, Orlando, FL*. 2009.

[27] Muthanna, C. and W.J. Devenport, Wake of a compressor cascade with tip gap, part 1: Mean flow and turbulence structure. AIAA journal, 2004. **42**(11): p. 2320-2331.

## Biography

My name is Keyvan Shaabani Lakeh. I study PhD in Internal Combustion Engine Group, at Department of Energy in Politecnico di Milano. My research interest is about turbocharging system study through numerical modeling.